

AIM Thermal Stresses in a Bar - Validation

Author(s): Sebastian Vecchi, ANSYS Inc.

Problem Specification

1. Pre-Analysis & Start-Up
2. Geometry
3. Mesh
4. Physics Setup
5. Results
6. Verification & Validation

Validation

In the pre-analysis, the maximum stress was calculated. In order to verify that our simulation was accurate, a comparison must be made. In order to calculate the maximum stress of the simulation, click on **Add next to Results**, and select **Calculated Value**. In the function drop down menu, select **Maximum**. Under the **Variable** drop down menu, select **Equivalent Stress** then press **Evaluate**. Very quickly, the maximum stress of the heated bar is calculated and displayed. This maximum value can also be seen in the panel on the left when the **Equivalent Stress 1** contour is displayed.

The table below compares the calculated and simulated values for maximum stress in a heated bar. With a difference of much less than 5%, we can consider our simulation to be accurate.

Calculated Value	Simulated Value	Difference
160.03 Mpa	160 Mpa	0.018%

[Go to all ANSYS AIM Learning Modules](#)